# NC Programming Codes

## NC Programming as per ISO (DIN 66025) and RS274

G-Codes simple definition

G00 Rapid traverse

G01 Linear interpolation with feedrate

G02 Circular interpolation (clockwise)

G03 Circular interpolation (counter clockwise)

G2/G3 Helical interpolation

G04 Dwell time in milliseconds

G05 Spline definition

G06 Spline interpolation

G07 Tangential circular interpolation / Helix interpolation / Polygon interpolation / Feedrate interpolation

G08 Ramping function at block transition / Look ahead "off"

G09 No ramping function at block transition / Look ahead "on"

G10 Stop dynamic block preprocessing

G11 Stop interpolation during block preprocessing

G12 Circular interpolation (cw) with radius

G13 Circular interpolation (ccw) with radius

G14 Polar coordinate programming, absolute

G15 Polar coordinate programming, relative

G16 Definition of the pole point of the polar coordinate system

G17 Selection of the X, Y plane

G18 Selection of the Z, X plane

G19 Selection of the Y, Z plane

G20 Selection of a freely definable plane

G21 Parallel axes "on"

G22 Parallel axes "off"

G24 Safe zone programming; lower limit values

G25 Safe zone programming; upper limit values

G26 Safe zone programming "off"

G27 Safe zone programming "on"

G33 Thread cutting with constant pitch

G34 Thread cutting with dynamic pitch

G35 Oscillation configuration

G38 Mirror imaging "on"

G39 Mirror imaging "off"

G40 Path compensations "off"

G41 Path compensation left of the work piece contour

G42 Path compensation right of the work piece contour

G43 Path compensation left of the work piece contour with altered approach

G44 Path compensation right of the work piece contour with altered approach

G50 Scaling

G51 Part rotation; programming in degrees

G52 Part rotation; programming in radians

G53 Zero offset off

G54 Zero offset #1

G55 Zero offset #2

G56 Zero offset #3

G57 Zero offset #4

G58 Zero offset #5

G59 Zero offset #6

G63 Feed / spindle override not active

G66 Feed / spindle override active

G70 Inch format active

G71 Metric format active

G72 Interpolation with precision stop "off"

G73 Interpolation with precision stop "on"

G74 Move to home position

G75 Curvature function activation

G76 Curvature acceleration limit

G78 Normalcy function "on" (rotational axis orientation)

G79 Normalcy function "off"

G80 - G89 for milling applications:

G80 Canned cycle "off"

G81 Drilling to final depth canned cycle

G82 Spot facing with dwell time canned cycle

G83 Deep hole drilling canned cycle

G84 Tapping or Thread cutting with balanced chuck canned cycle

G85 Reaming canned cycle

G86 Boring canned cycle

G87 Reaming with measuring stop canned cycle

G88 Boring with spindle stop canned cycle

G89 Boring with intermediate stop canned cycle

G81 - G88 for cylindrical grinding applications:

G81 Reciprocation without plunge

G82 Incremental face grinding

G83 Incremental plunge grinding

G84 Multi-pass face grinding

G85 Multi-pass diameter grinding

G86 Shoulder grinding

G87 Shoulder grinding with face plunge

G88 Shoulder grinding with diameter plunge

G90 Absolute programming

G91 Incremental programming

G92 Position preset

G93 Constant tool circumference velocity "on" (grinding wheel)

G94 Feed in mm / min (or inch / min)

G95 Feed per revolution (mm / rev or inch / rev)

G96 Constant cutting speed "on"

G97 Constant cutting speed "off"

G98 Positioning axis signal to PLC

G99 Axis offset

G100 Polar transformation "off"

G101 Polar transformation "on"

G102 Cylinder barrel transformation "on"; Cartesian coordinate system

G103 Cylinder barrel transformation "on," with real-time-radius compensation (RRC)

G104 Cylinder barrel transformation with center line migration (CLM) and RRC

G105 Polar transformation "on" with polar axis selections

G106 Cylinder barrel transformation "on" polar-/cylinder-coordinates

G107 Cylinder barrel transformation "on" polar-/cylinder-coordinates with RRC

G108 Cylinder barrel transformation polar-/cylinder-coordinates with CLM and RRC

G109 Axis transformation programming of the tool depth

G110 Power control axis selection/channel 1

G111 Power control pre-selection V1, F1, T1/channel 1 (Voltage, Frequency, Time)

G112 Power control pre-selection V2, F2, T2/channel 1

G113 Power control pre-selection V3, F3, T3/channel 1

G114 Power control pre-selection T4/channel 1

G115 Power control pre-selection T5/channel 1

G116 Power control pre-selection T6/pulsing output

G117 Power control pre-selection T7/pulsing output

G120 Axis transformation; orientation changing of the linear interpolation rotary axis

G121 Axis transformation; orientation change in a plane

G125 Electronic gear box; plain teeth

G126 Electronic gear box; helical gearing, axial

G127 Electronic gear box; helical gearing, tangential

G128 Electronic gear box; helical gearing, diagonal

G130 Axis transformation; programming of the type of the orientation change

G131 Axis transformation; programming of the type of the orientation change

G132 Axis transformation; programming of the type of the orientation change

G133 Zero lag thread cutting "on"

G134 Zero lag thread cutting "off"

G140 Axis transformation; orientation designation work piece fixed coordinates

G141 Axis transformation; orientation designation active coordinates

G160 ART activation

G161 ART learning function for velocity factors "on"

G162 ART learning function deactivation

G163 ART learning function for acceleration factors

G164 ART learning function for acceleration changing

G165 Command filter "on"

G166 Command filter "off"

G170 Digital measuring signals; block transfer with hard stop

G171 Digital measuring signals; block transfer without hard stop

G172 Digital measuring signals; block transfer with smooth stop

G175 SERCOS-identification number "write"

G176 SERCOS-identification number "read"

G180 Axis transformation "off"

G181 Axis transformation "on" with not rotated coordinate system

G182 Axis transformation "on" with rotated / displaced coordinate system

G183 Axis transformation; definition of the coordinate system

G184 Axis transformation; programming tool dimensions

G186 Look ahead; corner acceleration; circle tolerance

G188 Activation of the positioning axes

G190 Diameter programming deactivation

G191 Diameter programming "on" and display of the contact point

G192 Diameter programming; only display contact point diameter

G193 Diameter programming; only display contact point actual axes center point

G200 Corner smoothing "off"

G201 Corner smoothing "on" with defined radius

G202 Corner smoothing "on" with defined corner tolerance

G203 Corner smoothing with defined radius up to maximum tolerance

G210 Power control axis selection/Channel 2

G211 Power control pre-selection V1, F1, T1/Channel 2

G212 Power control pre-selection V2, F2, T2/Channel 2

G213 Power control pre-selection V3, F3, T3/Channel 2

G214 Power control pre-selection T4/Channel 2

G215 Power control pre-selection T5/Channel 2

G216 Power control pre-selection T6/pulsing output/Channel 2

G217 Power control pre-selection T7/pulsing output/Channel 2

G220 Angled wheel transformation "off"

G221 Angled wheel transformation "on"

G222 Angled wheel transformation "on" but angled wheel moves before others

G223 Angled wheel transformation "on" but angled wheel moves after others

G265 Distance regulation – axis selection

G270 Turning finishing cycle

G271 Stock removal in turning

G272 Stock removal in facing

G274 Peck finishing cycle

G275 Outer diameter / internal diameter turning cycle

G276 Multiple pass threading cycle

G310 Power control axes selection /channel 3

G311 Power control pre-selection V1, F1, T1/channel 3

G312 Power control pre-selection V2, F2, T2/channel 3

G313 Power control pre-selection V3, F3, T3/channel 3

G314 Power control pre-selection T4/channel 3

G315 Power control pre-selection T5/channel 3

G316 Power control pre-selection T6/pulsing output/Channel 3

G317 Power control pre-selection T7/pulsing output/Channel 3

Note that some of the above G-codes are not standard. Specific control features, such as laser power control, enable those optional codes.

## M codes simple definition

| | |
|---|---|
| M00 | Unconditional stop |
| M01 | Conditional stop |
| M02 | End of program |
| M03 | Spindle clockwise |
| M04 | Spindle counterclockwise |
| M05 | Spindle stop |
| M06 | Tool change (see Note below) |
| M19 | Spindle orientation |
| M20 | Start oscillation (configured by G35) |
| M21 | End oscillation |
| M30 | End of program |
| M40 | Automatic spindle gear range selection |
| M41 | Spindle gear transmission step 1 |
| M42 | Spindle gear transmission step 2 |

M43 Spindle gear transmission step 3

M44 Spindle gear transmission step 4

M45 Spindle gear transmission step 5

M46 Spindle gear transmission step 6

M70 Spline definition, beginning and end curve 0

M71 Spline definition, beginning tangential, end curve 0

M72 Spline definition, beginning curve 0, end tangential

M73 Spline definition, beginning and end tangential

M80 Delete rest of distance using probe function, from axis measuring input

M81 Drive On application block (resynchronize axis position via PLC signal during the block)

M101-M108 Turn off fast output byte bit 1 (to 8)

M109 Turn off all (8) bits in the fast output byte

M111-M118 Turn on fast output byte bit 1 (to 8)

M121-M128 Pulsate (on/off) fast output byte bit 1 (to 8)

M140 Distance regulation “on” (configured by G265)

M141 Distance regulation “off”

M150 Delete rest of distance using probe function, for a probe input (one of 16, M151-M168)

M151-M158 Digital input byte 1 bit 1 (to bit 8) is the active probe input

M159 PLC cannot define the bit mask for the probe inputs

M160 PLC can define the bit mask for the probe inputs (up to 16)

M161-M168 Digital input byte 2 bit 1 (to bit 8) is the active probe input

M170 Continue the block processing look ahead of the part program (cancel the M171)

M171 Stop the block processing look ahead of the probe input part program segment (like a G10)

M200 Activate the hand wheel operation in the automatic mode (to introduce an offset in the program)

M201-M208 Select the axis (by number from 1 to 8) for the hand wheel operation

M209 Activate the hand wheel operation in the automatic mode, with PLC control of the axis selection

M210 Deactivate the hand wheel input while in the automatic mode

M211 Deactivate this hand wheel feature and also remove the hand wheel offset (if any)

M213 Spindle 2 clockwise

M214 Spindle 2 counterclockwise

M215 Spindle 2 stop

M280 Switchable spindle/rotary axis, rotary axis on, first combination

M281 Switchable spindle/rotary axis, rotary axis on, second combination

M290 Switchable spindle/rotary axis, spindle enabled, first combination

M291 Switchable spindle/rotary axis, spindle enabled, second combination

Note: Other machine functions, like tool change (usually M06) or coolant control, have their M-code value specified by the PLC application not by the CNC software. Most of the M-code values in above list are configurable.

Other M-codes (up to M699) can be handled by the PLC application based on the particular machine requirements.